Snap-Fit Optimization for Achieving Desired Insertion and Retention Forces

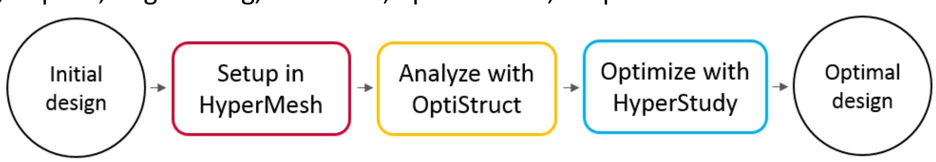
Eamon Whalen, Ravi Kodwani
Altair Engineering Inc.
Revised 6/25/2018

Abstract

Snap-fits are ubiquitous engineering features used to quickly and inexpensively assemble plastic parts. The geometric, material, and contact nonlinearities associated with snap-fit problems can present modeling challenges. Quasi-static solutions with explicit solvers are commonly used to analyze snap-fits; however, OptiStruct's nonlinear solver now possess the ability to solve these highly nonlinear problems implicitly. The first part of this study discusses an effective approach to using OptiStruct for the implicit finite element analysis of snap-fits. Once an accurate simulation model has been created, engineers typically make design changes in order to achieve desired insertion and retention forces. The second part of this study details how HyperMesh morphing and HyperStudy can be used to optimize the snap-fit design, resulting in desired insertion and retention forces while minimizing mass and ensuring structural integrity. The approach documented in this report can reduce the design time, material use, and failure rate of snap-fits used in industry.

Keywords: contact, implicit, large sliding, nonlinear, optimization, snap-fit

Workflow:



Introduction

A snap-fit is a type of formfitting joint which is typically molded directly into a plastic part. Although the term "snap-fit" is used to refer to a variety of different joints, all snap-fits consist of some type of protruding component which elastically deforms during the joining operation before snapping into an undercut or depression [1]. A snap-fit may be designed to be separable or inseparable, depending on the application. Cantilever snap-fits consist of a cantilever arm with a hook on the end that deflects during insertion. Other types of snap-fits include annular snap-fits, which consist of the mating of concave and convex surfaces, and torsion snap-fits, where deflection occurs as a result of a twisting motion.

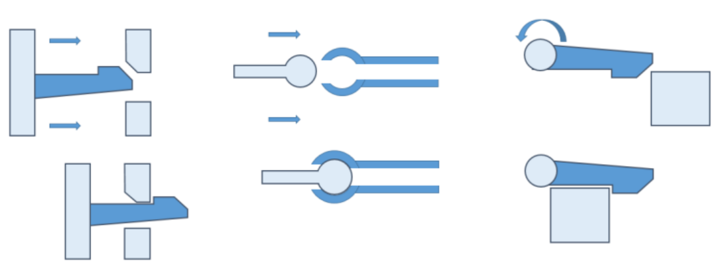


Figure 1: Cantilever snaps-fits (left), annular snap-fits (center), and torsion snap-fits (right) are among the three most common snap-fit joints used in industry [1].

Although snap-fits have existed for many years, their use in automotive engineering has recently increased for a number of reasons [2]. Snap-fit joints eliminate the need for a fastener, reducing the cost and weight associated with joining plastic parts. Increased emphasis on Design for Assembly (DFA) favors snap-fits because they can be assembled in less time and with less ergonomic strain than other fasteners [3]. Snap-fits allow users to quickly disassemble parts of dissimilar materials, aiding in the recycling process [4]. Lastly, the development of polymer technology and composite materials has allowed snap-fits to be used for heavier applications requiring larger retention forces. Some common examples of snap-fits used in automotive engineering include air filter housings, throttle bodies, temperature and pressure sensors, electrical connectors, and engine intake manifolds [2].

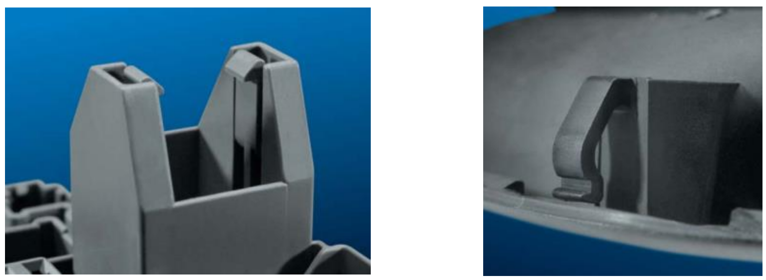


Figure 2: Snap-fit joints are commonly used in automotive engineering because they can be used to quickly and inexpensively assemble plastic parts. Pictured: an automotive fuse box (left) and door handle bezel (right) (Photos taken from BASF "*Snap-Fit Design Manual*" [2]).

Snap-fit Design and Analysis

A number of factors must be considered during the snap-fit design process. The insertion force is defined as the maximum force required by the user to fully join the snap-fit. Similarly, the retention force, or extraction force, is the maximum force required to separate the snap-fit joint. Designers typically aim to reduce the insertion force below an ergonomically defined limit while maintaining the retention force required in the design application. Insertion and retention forces that are too high will be difficult or impossible for a human to apply and could cause injury. Retention forces that are too low could lead to the unintended separation of the joint. A designer should also consider whether the presence of any plastic deformation during insertion will affect the joint's performance. Other considerations, such as the effects of tolerance on joint play, creep, and multicycle integrity may also be considered, and are highly dependent on the application [5].

With the increased use of snap-fits has come a larger demand to accurately and efficiently model their performance. The Finite Element Analysis (FEA) of snap-fit joints is highly nonlinear due to the effects of contact, large displacement, and material nonlinearity. Such problems are commonly solved using explicit solvers as they are more robust against nonlinearities; however, the computational resources required to perform explicit analysis are often extensive [6] [7]. Implicit modeling has several advantages, namely the lack of small time step requirements. The implicit method may also one day lead to the ability to perform in-solver optimization. In this study, Altair's OptiStruct nonlinear solver is used to implicitly analyze a generic snap-fit problem. Shape variables are then defined which can be altered to generate new designs. Finally, an optimization algorithm in HyperStudy is used to manipulate these shape variables until a design has been achieved that satisfies the desired insertion forces, retention forces, and plastic strain limit.

Model Preparation in HyperMesh

The model used in this study is a simple representation of a cantilever hook snap-fit inserting into a flexible slot. A prescribed displacement is applied to the base of the cantilever hook and the base of the slot is fixed in space. As the snap-fit design is symmetric about a center plane, only half of the snap-fit is included in the model. The following section describes how HyperMesh and OptiStruct were used to prepare and solve the model.

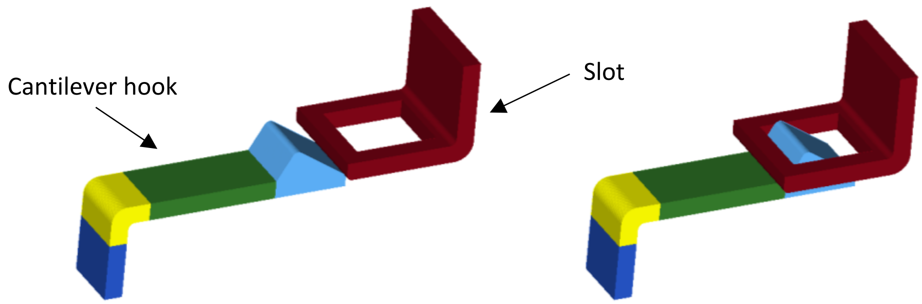
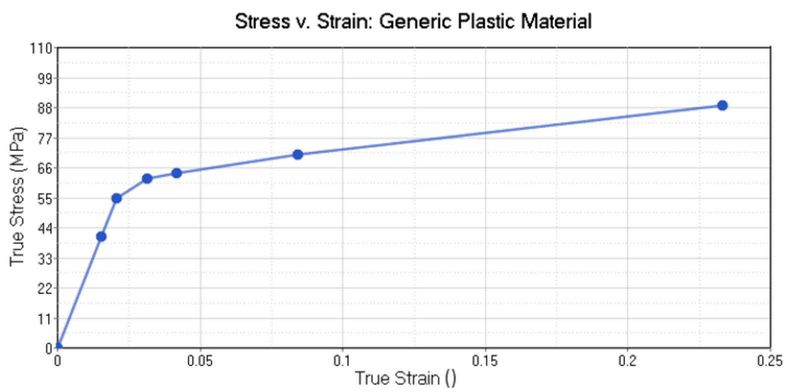


Figure 3: A simple cantilever snap-fit was used as the benchmark model for this study. The model contains contact, large displacement, and material nonlinearity.

Material Definition

Both components in this study have been assigned a representative plastic material. The material behavior is defined by a density, Young's Modulus, Poisson's Ratio, and stress versus plastic strain dataset.



Young's Modulus	2700 MPa
Poison's Ratio	0.35
Density	1.04 e-9 Mg/mm³

Figure 4: A piecewise linear plastic material definition was used to define snap-fit model material. Note that the stress v. strain data used in this report is for example only and does not represent actual test data.

A HyperMesh MAT1 card was used to define the material, and a TABLES1 load collector was used to define the stress versus strain data.

Meshing

The model used in this study involves simple geometry which could easily be meshed with hexahedron elements; however, second order tetrahedron elements were used since they are most common in industry. A more refined mesh was used at the fillet section of the hook component where stress concentrations are expected to occur. The resulting mesh contains 7,282 elements and 13,194 nodes.

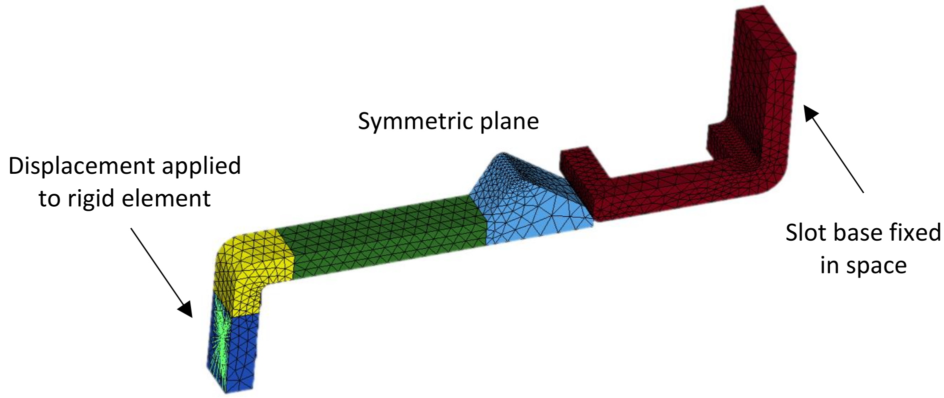


Figure 5: Second order tetrahedron elements were used to model the snap-fit because they can easily mesh a wide array of complex geometries. The boundary conditions are summarized above.

Boundary Conditions

SPC constraints were created to a) fix the bottom end of the slot in space, b) maintain symmetry constraints along the cut plane of both parts, and c) apply a prescribed displacement of 7.8 mm to the independent node of a rigid element that joins all nodes on the bottom surface of the snap-fit. The use of a rigid element simplifies post-processing because reaction forces can be queried at a single point. 7.8 mm is the distance that the snap must travel to completely join the joint.

Contact

Two contact surfaces were defined, one on each part. The hook surface is defined as the master and slot surface is defined to be the slave. Surface to surface contact is defined between the two contact surfaces. Note that this model features small fillets on contact edges rather than sharp edges. A smooth transition between surfaces makes it easier for the solution to converge. FINITE sliding is applied because the relative displacement is larger than the average element length. A kinetic frictional coefficient of 0.15 is applied to the surface. To aid in convergence, contact stabilization is applied to the contact. The OptiStruct card CNTSTB was used to specify contact stabilization parameters.

Solver Parameters

Three load collectors are used to define the input parameters for OptiStruct’s nonlinear solver. The first and only required load collector for nonlinear analysis has the card image NLPARM. This load collector is used to specify the number of implicit load sub-increments and the maximum number of iterations per increment. Convergence will fail once the maximum number of iterations is reached. The NLADAPT load collector is used to define limits for the largest and smallest time increment allowed during the Newton-Raphson method. Setting DTMAX = 0.01 ensures the model will be solved in no less than 100 time increments. The NLOUT load collector specifies the maximum number of intermediate increments to be output but does not affect the results of the simulation. Without defining the NLOUT card, the solver would only output results for the model at the end of the simulation. The PARAM, LGDISP control card is used to activate large displacement nonlinear quasi-static analysis. The presence of large sliding requires that large displacement theory to be used in order to obtain accurate results.

Table 1: Various load collector cards used to define the analysis parameters. The * denotes values which were left at their default values.

Card	Variable	Meaning	Value
CNTSTB	S0	Stabilization scale factor at the beginning of the subcase	1.0
	S1	Stabilization scale factor at the end of the subcase	1e-4
	SCALE	Stabilization coefficient scale factor	1
	TFRAC	Scale factor in the tangential direction with respect to the normal direction	0.2
NLPARM	NINC	Number of implicit load sub-increments	10 *
	MAXITER	Limit on number of implicit iterations for each load increment	25 *
NLADAPT	DTMAX	Maximum allowable time increment	0.01
	DTMIN	Minimum allowable time increment	1e-6
NLOUT	NINT	Number of intervals specified to output intermediate results	100

The analysis is defined by two nonlinear quasi-static load steps: one for insertion and one for extraction. The Continue Nonlinear Subcase Parameter (CNTNLSUB) is used to ensure the extraction subcase begins at the end of the insertion subcase, thus preserving any plastic strains and deformations that may have occurred during insertion. As the insertion and retention forces are of interest in this study, the SPCF output parameter was activated inside each loadstep. This parameter tells OptiStruct to output the reaction forces at each SPC constraint. Similarly, the STRAIN output request is activated for elements in the cantilever beam, which are considered critical because they are most likely to yield.

OptiStruct Analysis Results

The model was analyzed using OptiStruct's nonlinear quasi-static solver. The total runtime was just under 49 minutes using four CPUs. HyperView was used to display the animation and SPCF forces. Convergence was achieved and the resulting animation depicts a smooth insertion of the snap-fit, followed by a complete extraction. The SPCF forces in the Z-axis (direction of prescribed motion) represent the snap-fit insertion and extraction forces. The insertion force and extraction force were found to be 140 N and 170 N respectively.

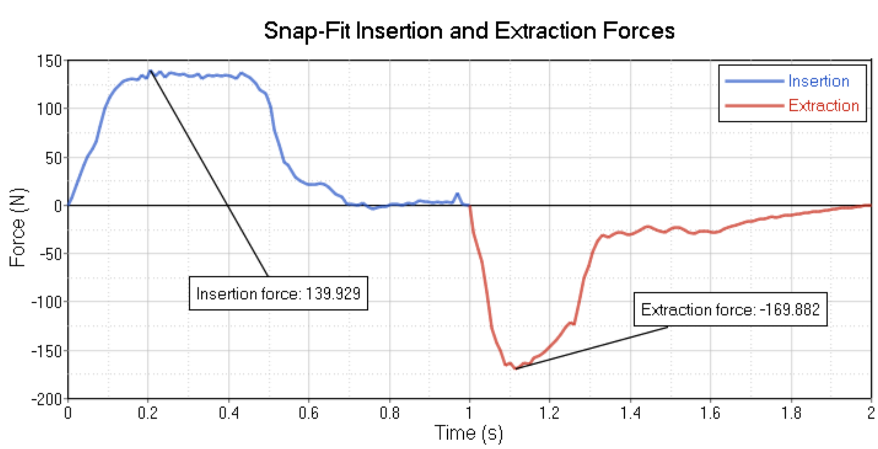


Figure 6: The insertion and extraction forces can be viewed by plotting the reaction forces at the constraint where the prescribed displacement is applied. The force plot above is typical for a cantilever snap-fit design.

The HyperView post processor was also used to generate a plastic strain contour. Note that plastic strains were only requested for elements determined to be critical. One plastic region has formed in the model at the inner fillet of the snap-fit shaft. The largest elemental plastic strain was found to be 5.7×10^{-5} which can be considered negligible.

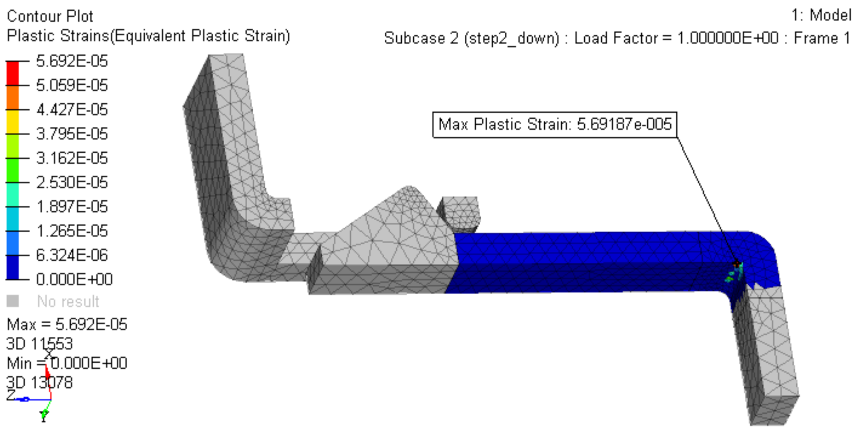


Figure 7: Plastic strains on the order of 10^{-5} are observed at the corner of the cantilever arm. Strains of this magnitude can be considered negligible.

Optimization Setup in HyperStudy

Once the existing snap-fit model has been analyzed the designer must decide whether its performance is satisfactory. Commonly, the insertion and extraction forces predicted by analysis do not fall within the desired range, thus requiring small design changes to be made to the snap-fit. Further design changes may be necessary to reduce plastic strain in the model. Once these changes have been applied by a designer, the new snap-fit design is reanalyzed to see whether the desired performance has been met. Design optimization can be used to automate the design/analysis loop and select features most likely to satisfy a given set of constraints.

Design Variables

Shape optimization was selected for this study because it allows the designer to predefine a few shape variables based on the original design. These are likely the same shape variables that a designer would edit manually when attempting to improve the part. Three shape variables were defined using the morphing tool in HyperMesh. The variables are the insertion face angle, the retention face angle, and the shaft taper angle. Since extreme morphing can cause elements to become distorted, the element quality check parameter in OptiStruct was turned off for optimization (CHECKEL = NO). An alternative option would have been to re-mesh the morphed geometry before each iteration.

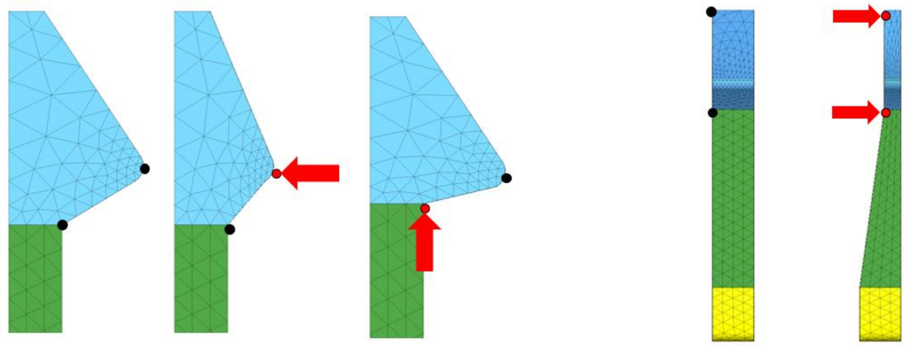


Figure 8: Three shape variables were defined in HyperMesh using morphing. These variables will be modified by the optimization algorithm until an optimal design is achieved. From left to right: the nominal design, alteration of the insertion face angle (SV #1), alteration of the retention face angle (SV #2), alteration of the cantilever taper angle (SV #3).

When this study was completed OptiStruct did not support in-solver optimization for large displacement problems or problems with material nonlinearity. However, HyperStudy is a design exploration tool which can be used to perform optimization by iteratively creating OptiStruct models and analyzing the results. The three shape variables defined in HyperMesh were imported as HyperStudy variables. The responses were defined to be a) the mass of the male snap-fit, b) the insertion force, c) the extraction force, and d) the maximum plastic strain at the end of simulation.

Design Constraints

Design constraints are applied to responses in order to ensure that the resulting model meets all design criteria. For this study, the insertion force should fall in the range of 125-135 N and the extraction force should fall in the range of 180-190 N. To prevent permanent deformation during operation, the maximum plastic strain in the model should not exceed 1×10^{-3}. Note that the nominal model violates the insertion and extraction constraints but satisfies the plastic strain constraint.

Responses

The optimization objective is to minimize the mass of the cantilever snap-fit component. The purpose of this study is not necessarily to reduce the mass of the snap-fit, as snap-fits typically represent a small fraction of the mass of a part. However there are plausibly an infinite number of designs which satisfy the given constraints. The addition of an objective ensures, of all the feasible designs, HyperStudy will select the one with the lowest mass.

Table 2: Model Responses

Variable	**Meaning**	**Initial Value**	**Constraint /Objective**
m	Mass of the cantilever snap-fit component (g)	7.62	Minimize m
i	The maximum SPCF force during snap-fit insertion (N)	139.9	$125 \le i \le 135$
e	The maximum SPCF force during snap-fit extraction (N)	-169.9	$-190 \le e \le -180$
s	Maximum plastic strain among critical elements at the end of extraction	5.69e-5	$s \le 0.001$

Optimization Algorithm

There are multiple optimization algorithms available in HyperStudy, each best suited for a particular type of problem. The Global Response Surface Method (GRSM) was selected for this study. GRSM is a powerful, general purpose optimization algorithm that combines response surface methods with global search [8]. The beginning of the optimization is devoted to performing a Design of Experiment (DOE) which is used to construct an initial response surface. At each subsequent iteration, the DOE is expanded to include the predicted optimum from the current response surface, as well as points taken from a global design space search. The later reduces the probability of getting stuck on local minima. GRSM does not have specific convergence criteria, rather, the algorithm runs for a specified number of evaluations. For this study GRSM was run for 75 evaluations.

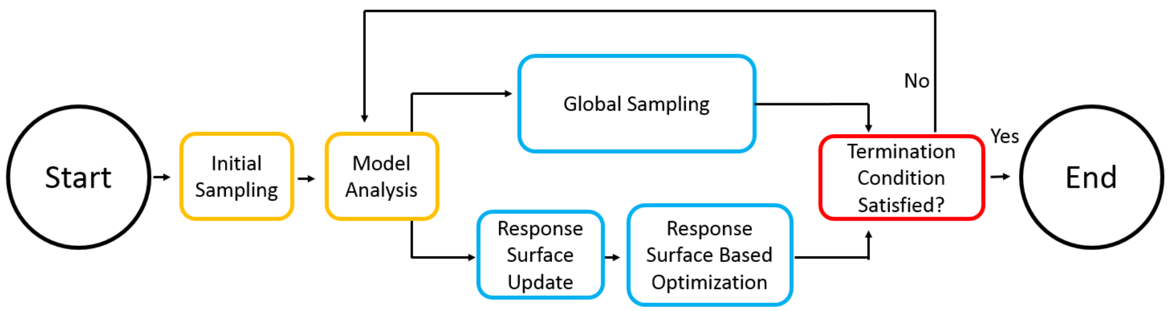


Figure 9: The Global Response Surface Method (GRSM) algorithm combines response surface-based optimization with global sampling to determine an optimal solution. In this study, the termination condition was a fixed number of evaluations. (Diagram adapted from "*Optimal Design Exploration Using Global Response Surface Method: Rail Crush*" by Joseph Pajot) [8].

Optimization Results

The optimization study completed 75 evaluations which were used to generate 30 distinct designs. The optimized design was reached after the 15th iteration.

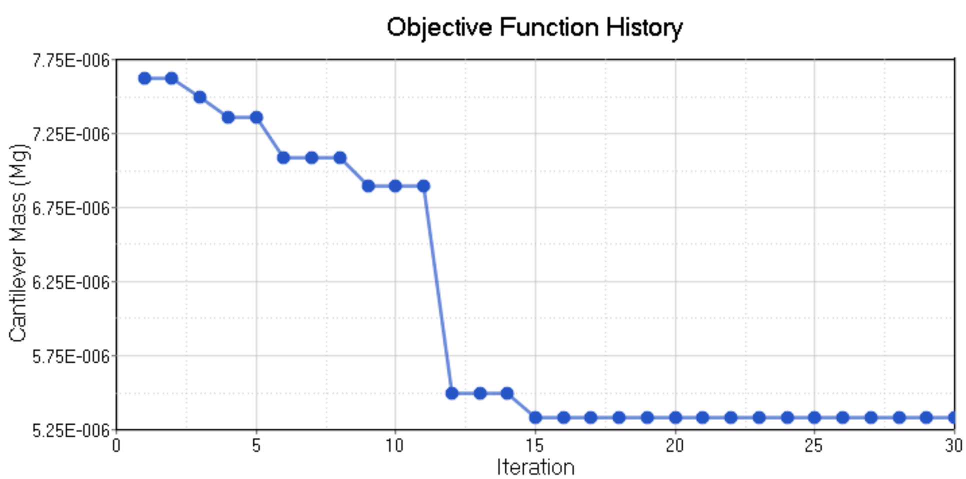


Figure 10: Each iteration of GRSM requires multiple solver evaluations. In this study, 75 evaluations were used to generate 30 distinct snap-fit designs. As the optimization progresses, the algorithm attempts to minimize the structure's mass while obeying the design constraints. The optimal structure was determined in the 15th iteration.

The resulting design has a slightly shallower insertion face, which is expected since the nominal insertion force exceeded the allowable range. Similarly, the new design has a steeper extraction face because the nominal extraction force was smaller in magnitude than the allowable range. The cantilever beam has been tapered to achieve a near constant strain across its length. The taper generated during optimization is similar to that which is recommended in most snap-fit design guides [1] [2].

Table 3: Design variables before and after optimization. Note that the value of a shape design variable represents the percentage of its originally morphed shape and should not be confused with a physical dimension.

Variable	Meaning	Initial Value	Final Value
ins_ang	Affects the insertion face angle	0	0.422
ext_ang	Affects the extraction face angle	0	0.734
taper_ang	Affects the shaft taper angle	0	0.941

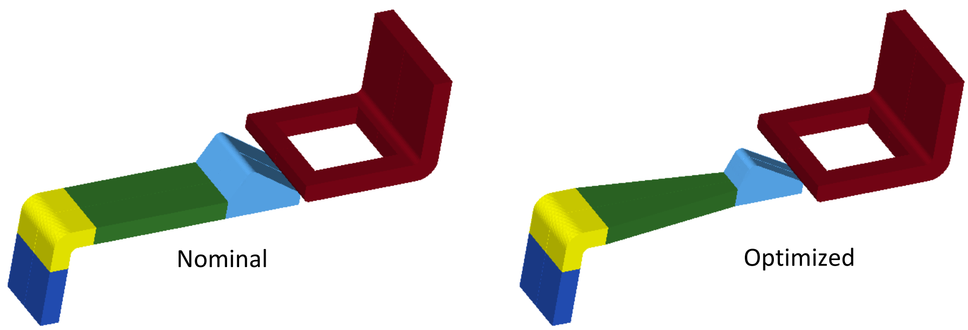


Figure 11: The nominal snap-fit design (left) has been optimized to achieve desired insertion forces and extraction forces (right). The optimized design's mass has been reduced by 30%. Note that results have been reflected about the symmetric plane in order to portray the full design.

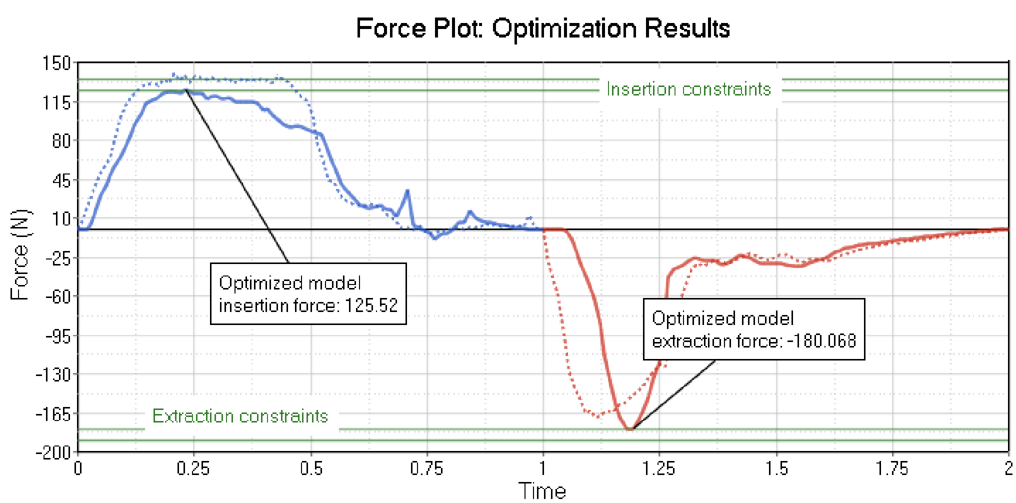


Figure 12: The optimized design exhibits insertion and extraction forces that lie within the desired ranges (solid lines). The dotted lines represent the insertion and extraction forces of the nominal model.

Table 4: Comparing the responses of the initial and final models.

		Initial Design		Final Design	
Response	**Acceptable Range**	**Value**	**Acceptable?**	**Value**	**Acceptable?**
Insertion force (N)	125 – 135	139.9	No	125.5	Yes
Extraction force (N)	-190 – -180	-169.9	No	-180.1	Yes
Maximum plastic strain	≤ 0.001	5.69-5	Yes	9.23e-4	Yes
Mass (g)	N/A	7.62	N/A	5.33	-30%

The resulting model satisfies all design requirements. Now that the optimization has been setup for this generic snap-fit, the design of a new snap-fit with different constraint values would be trivial. In this way, the study outlined in this report could serve as a template for optimizing any snap-fit of similar geometry.

Conclusion

This study outlines how snap-fits used in industry can be analyzed using OptiStruct's nonlinear implicit solver and how its performance can be optimized using HyperStudy. As the use of snap-fits in automotive engineering increases it becomes critical that their behavior can be accurately predicted with minimal computational resources. Implicit solvers now possess the ability to solve problems with contact, large displacement, and material nonlinearity. In this study, a cantilever snap-fit model was prepared using HyperMesh and analyzed with OptiStruct. It was determined that the insertion force was too large and that the extraction force was too small (in this case the force targets were arbitrary). Next, design optimization was performed on the model to achieve the desired insertion and extraction forces. HyperStudy was used to systematically generate several variations of the nominal design and test them to determine the optimal design. The result is a snap fit that satisfies the desired insertion and extraction forces, exhibits plastic strains below the specified tolerance, and uses 30% less material than the original design. Through design optimization, design engineers can reduce the design time, material use, and failure rate of snap-fits used in industry.

Acknowledgments

The authors would like to thank Girish Kalyan Mudigonda Kuravi and Rumpa Chandaroy for their expertise related to the OptiStruct solver, as well as Ramandeep Gill for his support of the HyperStudy application. Without their support, this study would not have been possible.

References

1. Bayer MaterialScience LLC, "Snap-Fit Joints for Plastic Parts," http://fab.cba.mit.edu/classes/S62.12/people/vernelle.noel/Plastic_Snap_fit_design.pdf, accessed Oct. 2016.

2. BASF, "Snap-Fit Design Manual," http://web.mit.edu/2.75/resources/random/Snap-Fit%20Design%20Manual.pdf, accessed Oct. 2016.

3. Dolah, M. and Rahman, A., "Design for Assembly Understanding to improve Malaysian Office's Furniture Assembly ergonomically," presented at PPCOE 2007, Bangkok, Thailand Oct 17-19, 2007.

4. Ruan, T., "Selection and Optimization of Snap-Fit Features via Web-Based Software," Ph.D. thesis, Mechanical Engineering Department, The Ohio State University, Columbus, 2005.

5. Lee, C., and BASF, "Improving Snapfit Design," http://knowledge.ulprospector.com/1248/pe-snapfit-3/, accessed Oct. 2016.

6. Kulkarni, V., Musale, G., Vallurupalli, S., "Effective Snap-Fit Design Using Radioss" http://www.altairatc.com/india/previous-events/2011/presenters/RNL_12_Effective_Snap-Fit_Design_Using_RADIOSS_Solver_TTL.pdf, accessed Oct. 2016

7. Abbey, T., "Nonlinear FE Analysis," National Agency for Finite Element Methods and Standards (NAFEMS) Training Course, Oct. 17 2016.

8. Pajot, J., "Optimal Design Exploration Using Global Response Surface Method: Rail Crush," http://www.altairhyperworks.com/ResourceLibrary.aspx?translation_id=2408, accessed Oct. 2016.